

AN EFFECT OF TIP CLEARANCE ON AERO PERFORMANCE IN AXIAL FLOW COMPRESSORS FOR AERO GAS TURBINE ENGINES

MANJUNATH S. DALBANJAN & NIRANJAN SARANGI

Gas Turbine Research Establishment, Bangalore, Karnataka State, India

ABSTRACT

Gas turbine engines have dominated aviation industry, because they are highly efficient and reliable. Axial flow compressor is a critical part which plays a crucial role in efficiently working of an aero gas turbine engine. The entry and exit of the working fluid is in axial direction with respect to rotational axis in axial flow compressor. Higher mass flow swallowing capacity and higher efficiency for a small frontal area are unique features of axial flow compressors making an ideal option for the gas turbines used in jet engines. One of the major source of performance deterioration/enhancement in compressors is because of tip clearance between the rotor blade tip and casing. The tip clearance causes leakage of flow. The leaked flow interacts with the core flow and the wall boundary layer and control aerothermodynamics behavior of flow in the tip regions. This tip leakage flow also has an impact on efficiency and operating envelope of the compressor. In this work, a steady state analysis has been carried out to understand the influence of tip clearance on the aerodynamic performance of transonic single stage wide chord blade axial flow compressor stage through three dimensional viscous analysis using ANSYS CFX. The analysis was carried out for different tip clearance configuration varying from 0% to 1.2% of rotor tip axial chord. The characteristics plots like corrected massflow rate versus pressure ratio and corrected massflow rate versus efficiency were plotted for various tip clearances at design speed. The generated database of tip clearance in this work will predict beforehand the extent of deterioration due to manufacturing deviation of the components on aerodynamic performance prior to engine testing in the test bed.

KEYWORDS: Axial Flow Compressor, Tip Clearance, CFD, Pressure Ratio, Surge Margin, Efficiency, Operating Range & Performance Characteristics

Received: May 06, 2019; **Accepted:** Jun 17, 2019; **Published:** Jul 10, 2019; **Paper Id.:** IJMPERDAUG201977

INTRODUCTION

Compressor is the critical component in gas turbine engine and the operating envelope of the gas turbine engines depends on the performance of the compressor. Due to unfavorable pressure gradient the flow in compressor is unsteady which causes flow separations resulting in rotating stall or surge to occur, which leads to compression system failure. Rotating stall or surge creates aerodynamic unsteadiness which imposes design restrictions in terms of surge margin and even on performance of compressor and gas turbine also. To achieve a wide operating range and maximizing the efficiency of single stage or multistage compressor poses a challenging task to the designer. The aero-thermal design dictates the efficiency and surge margin of an axial flow compressor. Axial flow compressor has an operating range, defined by a higher point called stall point and lower point called choke point for a specific speed, within which the compressor can operate efficiently. Stall point is the point at which aerodynamic instabilities will be initiated. At speeds below the design point both efficiency and aerodynamic stability gets affected because of the rotor tip gap variations and boundary wall [1-2]. Xicheng et. al.

[3] studied different tip gaps like constant and linearly variable tip gaps by using numerical methods. The results show the highest efficiency is not achieved with zero tip clearance. Boundary layer effect on a rotor stator stage axial flow compressor for two conditions i. e. one at design point and another at choke point were studied by Wang Songtao [4]. The study showed at the design condition, the circulation due to the tip flow travels through blade passage and impinges on the adjacent blade and tip vortex travels downstream along streamwise direction at the choke condition. Gong et. al. [5] showed that there is a flow turbulence due to tip leakage, and there is no leakage vortex appeared behind trailing edge due to reduction in the flow velocity. John et. al. [6] utilized DPIV for flow rate measurement in a multi stage axial flow compressor. The study showed that at 50% rotor chord there exists a reverse flow zone on the suction surface of the blade. Donghyun et. al. [7] developed a methodology to study the unsteady flow behaviour due to blade tip leakage. Experimental and numerical techniques were used to understand the rotor blade tip flow phenomena in the space between blade tip and casing zone in axial flow compressor by L. Suder [8], the study showed tip clearance effect causes separation of flow from tip to hub which will be at 75% of rotor span at 100% speed. Flow field gets periodically modified because of unsteady flows due to the relative motions between static and rotating parts [9]. Most of the studies are limited to design condition. In transonic compressor, at design condition flow is transonic and at well below design condition, flow is subsonic. Most of the studies are carried out on research compressor and not on compressor designed for field application. It is very essential to cover all these aspects of flow region to arrive at optimum clearances considering all the regions of flow encountered in entire operating zone of compressor designed for field application.

METHODOLOGY

The aerodynamic analysis process is a crucial part in the multidisciplinary development of an aero gas turbine engine compressor. The analysis will be performed to study the tip clearance effect on axial flow compressors, of aero gas turbine, engine by using commercially available software package 3-D Computational Fluid Dynamics (CFD) code. The methodology followed for the analysis process is explained below.

CFD Code Validation Study

Computational fluid dynamics has emerged as a valuable tool to achieve several design goals in turbomachinery applications. CFD is used for complete evaluation of various design modifications prior to testing. While CFD has been used intensively in turbomachinery applications, careful validation of the software is necessary hence an assessment of ANSYS-CFX code was carried out. NASA stage 37 by Reid and Moore [10], as a standard transonic compressor test case is considered here for the assessment of CFD tool. NASA stage 37 is inlet stage of an eight-stage core compressor, the design parameters are shown in table 1. The test results were used for comparison of predicted performance in our analysis. Detailed analyses were carried out on stage 37 model for 100% design speed at various flow points using $k-\omega$ turbulence model. The Stage overall characteristics were compared with test data by Reid and Moore [10]. It was observed that the efficiency was over predicted compare to NASA stage 37 test values but the pressure ratio was well within the range as tabulated in table 2.

Comparison of predicted and measured performance plots are shown in figures 1 & 2. The simulation result obtained using ANSYS-CFX code are compared to test data from published literature. Difference in simulation results and published test data is within 1%.

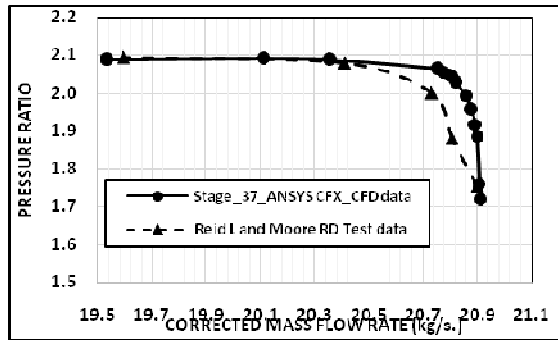


Figure 1: Massflow Rate v/s Pressure Ratio for NASA Stage 37

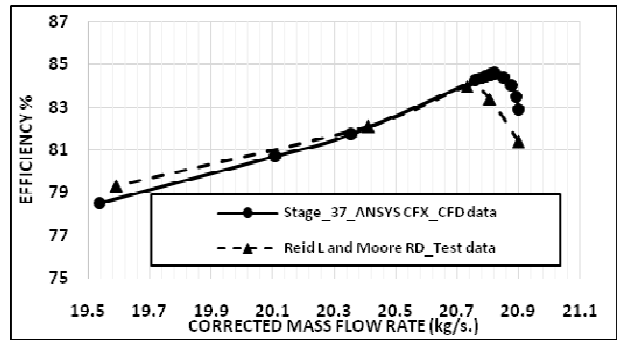


Figure.2 Massflow Rate v/s Efficiency for NASA Stage 37

Table 1: Rotor 37 Design Parameters

Parameter	Values
Design Pressure Ratio	2.106
Temperature Ratio	1.270
Efficiency	87.7%
Design Speed, RPM	17188.7
Mass flow, kg/s	20.188
mass flow_ Choke, kg/s	20.93
tip speedRotor, m/s	454.14
Hub to tip Ratio	0.70
Rotor blades	36 No. s
Type of, Blading	Multiple circular arc
Design tip clearance	0.2% span
Flow coefficient	0.453
Rotor Aspect Ratio	1.19

Table 2: Predictions of Performance Parameters of NASA Stage37

Parameter	Experimental Data (Reid L & Moore RD)	Simulation CFD Data (Ansys CFX)	Error (%)
Choke mass flow, kg/s	20.93	20.91	-0.10
Peak Efficiency %	84.00	84.56	0.67
Peak pressure rise	2.093	2.087	-0.29

Solid Modeling of Blade Geometry in Flow Domain

A 3D modelling tool UG-NX-8.5, was used to create a 3D solid model of a stage i. e. rotor and stator. A single blade passage was considered instead of full 360 degree model for flow analysis by applying periodic boundary condition it will drastically reduce the computational time and grid size. The performance parameters obtained with the single blade passage when multiplied with the number of blades of the rotor or stator will give total values of a stage. The flow domain is discretized into number of sub volumes for generation of hybrid grids.

Grid Generation

A 3D grid generation Ansys Turbogrid software was used for generation of grid for a single passage flow domain. For boundary layer resolving size functions were used. The appropriate boundary conditions were defined like at rotor inlet stagnation pressure and stagnation temperature and at back of stator static pressure at stator exit.

Grid Convergence Study

A mesh convergence analysis was performed by considering various mesh configuration at design speed of the axial flow compressor. A three dimensional viscous analysis was performed using 3D N-S equation solver ANSYS-CFX with H-type mesh topology for flow domain and O-grid mesh topology near blade surface were used and ensured that the boundary layer flow near the blade surface is captured in both rotor and stator domains. In tip region around 12 elements were created and with non matching grid topology in the tip region was used with proper alignment of interface. Mesh distribution was chosen based on prior design expertise and mesh sensitivity study. Mesh distribution of 95(streamwise)x40(circumferential)x40(spanwise) nodes were maintained for rotor and stator domain, near the blade surface and casing wall more dense grid, with y^+ value of 3 was maintained. A rotational periodic interface was used between rotor outlet and stator inlet domain which gives circumferentially averaged flow parameters. $k-\omega$ with shear stress transport turbulence viscosity model along with high resolution turbulence numerics and advection scheme was selected. Mesh for both rotor and stator stage domain were generated as shown in Figure 3. Grid sensitivity study was carried out to arrive at the final grid size. Four different cases by varying number of grids in rotor and stator blades were considered with performance parameters massflow and pressure ratio, as shown in Figure 4.

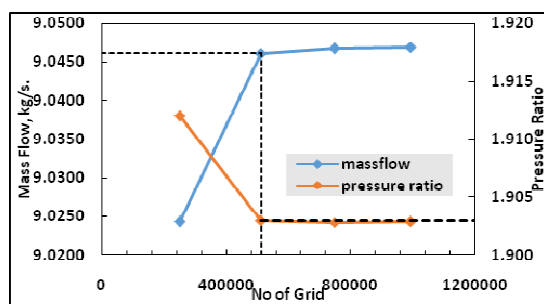


Figure 3: Mesh Domain for Rotor and Stator

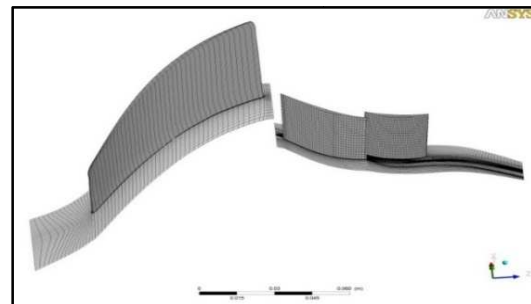


Figure 4: Grid Convergence Study

From the grid independence study it was observed that, there is no considerable variation between the mass flow and pressure ratio values, above 513000 grid size, so for further analysis the same grid configuration was considered.

3D Viscous Analysis

3-D steady N-S equations were solved using multi block grid with pressure based implicit solver ANSYS CFX. $k-\omega$ with shear stress transport (SST) turbulence model with high resolution advection option was chosen for the study.

Boundary Condition

Appropriate boundary conditions were applied for the flow domain, at inlet of rotor total pressure and total temperature. Interface mixing model between rotor and stator is considered as a stage where the flow parameters at the exit of rotor will be circumferentially averaged and distributed to the stator inlet at each iteration. At exit of stator static back pressure is applied and no-slip wall boundary condition was specified for blade, hub and casing.

Solution Methodology

The analysis was carried out for various tip clearance configuration varying from 0% to 1.2% of rotor tip axial chord at increment of 0.1%. The static back pressure at stator outlet was increased in steps to generate the constant speed characteristics. Near stall region the static pressure at the back of stator exit was raised in increment of 1 kPa to obtain the

converged solution. The point at which solution diverges is considered as stall point. Solution convergence was observed from RMS residuals for solving continuity, momentum and energy equations. Once the residuals drop below the convergence criteria the solution will stop. The residual convergence level for all the parameters is specified as 1×10^{-6} to get more accurate solution.

RESULTS AND DISCUSSIONS

The analysis was carried out and the constant speed characteristics were generated for various tip clearance configuration from 0% to 1.2% of rotor tip axial chord were obtained for 100% speed of the compressor.

Figure (5) shows the compressor performance at normalised design speed values (corrected mass flow rate versus total pressure ratio) for different tip clearance from zero % to 1.2 % clearance (i. e clearance gap = 1.2 % of rotor chord). It can be seen that the zero % clearance is giving highest mass flow but lower pressure ratio and hence lower surge margin of 8.6%. For tip clearance of 0.1% and 0.2% there is reduction in mass flow but the pressure ratio and surge margin are same as zero % clearance. For tip clearance of 0.3% the mass flow is lower by 0.4% but there is a gain in pressure ratio of 0.8% and surge margin obtained was 15.2%. When the tip clearance from 0.4% to 1.2% is increased it was observed that there is mass flow reduction and loss in the pressure ratio. Formation of tip leakage wakes at the tip area causing losses and consequently drop in pressure ratio of 1.5%.

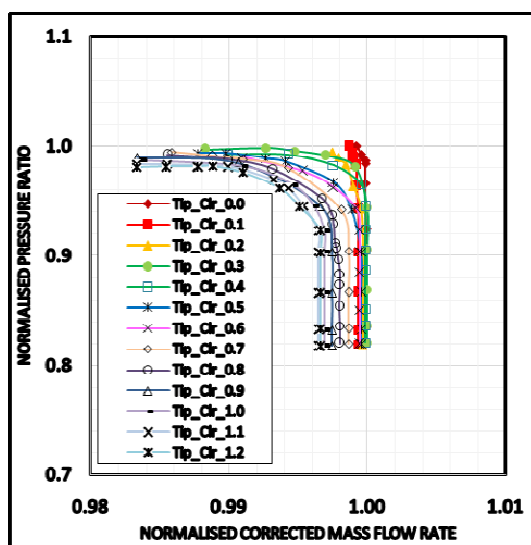


Figure 5: Normalised Corrected Massflow v/s Pressure Ratio

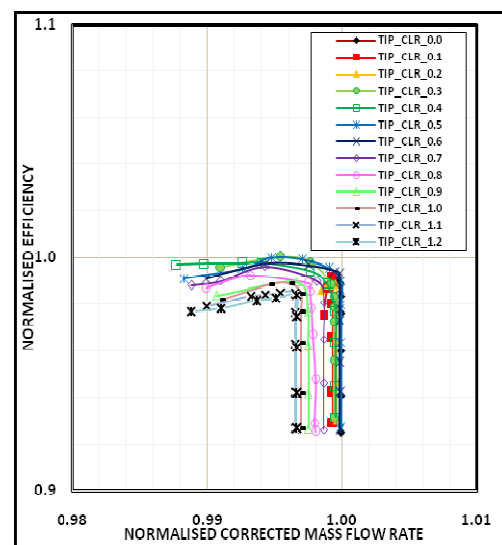


Figure 6: Normalised Corrected Massflow v/s Efficiency

Figure (6) shows the compressor performance at normalised design speed values (corrected mass flow rate versus efficiency) for tip clearance from zero % to 1.2 % (i. e clearance gap = 1.2 % of rotor chord). It can be seen that the zero % clearance is giving highest mass flow but lower efficiency this is due to the losses at the hub and tip because of formation of circular vortex in opposite direction to each other this loss is more predominant compare to tip leakage loss. For tip clearance of 0.1% and 0.2% the mass flow is reduced but the efficiency is same as zero tip clearance. For tip clearance of 0.3% the mass flow is lower but there is a gain in efficiency of 1.2%. As the tip clearance from 0.4% to 1.2% is increased it was observed that due to higher tip clearance the losses are high which resulted in loss in the mass flow and efficiency. It is also observed that because of tip leakage wakes form the tip zone there is a drop in efficiency of 2.1% from design value.

Figure (7) shows normalised efficiency versus tip clearance for a given total pressure ratio at three different condition of the compressor envelop i. e. Choke, design and the peak efficiency point. From the plot it was observed that for zero % to 0.3% tip clearance the efficiency is increasing and it has a peak value at 0.3% tip clearance at choke and design condition. Whereas the peak efficiency is at 0.5% tip clearance for stall condition. From 0.4 % to 1.2% of tip clearance the efficiency is dropping as the clearance is increased at design point and the same trend is observed at choke point. At stall condition from 0.9% to 1.2% tip clearance the efficiency is lower than the design point efficiency. This is due to tip leakage losses because of higher tip clearance.

Figure (8) shows the normalised total pressure ratio versus tip clearance for a given efficiency at three different conditions of the compressor envelop i. e. Choke, design and the stall point. From the plot it was observed that zero% to 0.2% tip clearance the pressure ratio is constant for all the three condition. The peak pressure ratio is achieved when the tip clearance is 0.3% and then the pressure ratio decreases with increase in tip clearance from 0.4% to 1.2% i. e. at design point and at stall point. The decrease in the pressure ratio is due to losses caused by formation of tip vortex at the tip region. At choke point from zero% to 1.2% tip clearance no significant variation in pressure ratio is observed.

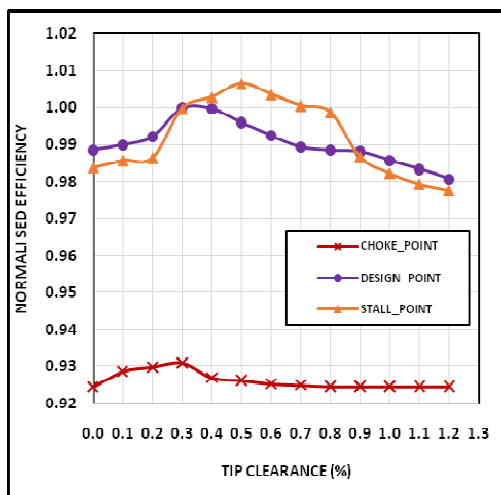


Figure 7: Normalised Efficiency v/s Tip Clearance

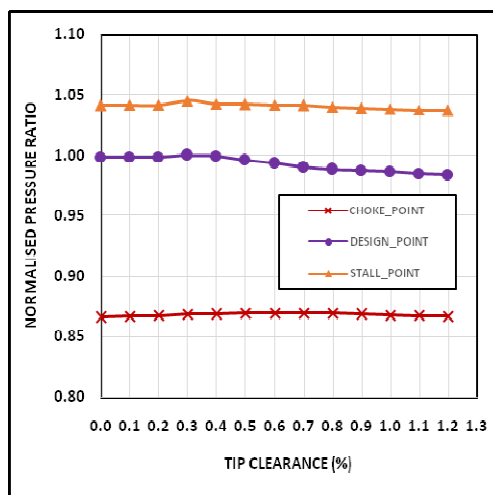


Figure 8: Normalised Total Pressure Ratio v/s Tip Clearance

Figure (9) shows the variation of total pressure in spanwise direction at exit of rotor at design point for the tip clearance of 0.3% and 1.2%. It is observed from the plot that at the tip zone there is a drop in total pressure of 8.5% when the tip clearance is increased from 0.3% to 1.2%.

Figure (10) plot shows spanwise variation of Mach number in relative terms at rotor exit at design point for the tip clearance of 0.3% and 1.2%. It is observed from the plot that at the tip Mach number shows higher value for 0.3% compare to 1.2% tip clearance.

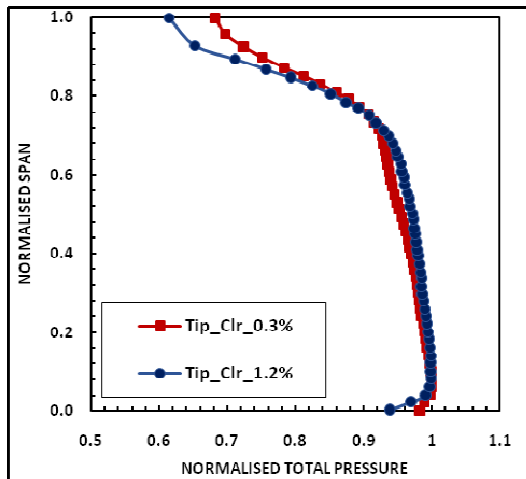


Figure 9: Spanwise Variation of Total Pressure at Rotor Exit

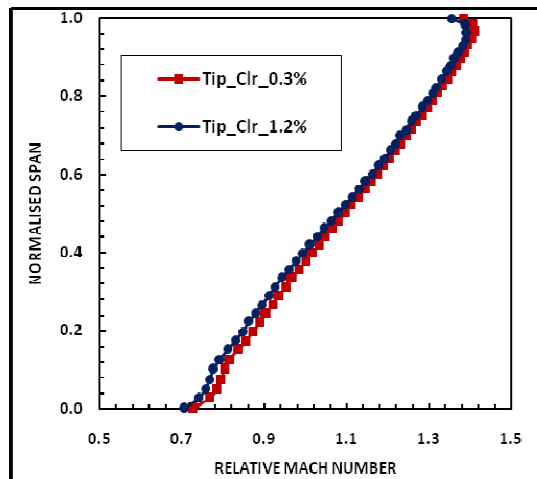


Figure 10: Spanwise Variation of Relative Mach number at Rotor Exit

CONCLUSIONS

- A3D viscous analysis is carried out to investigate the sensitivity of tip clearance on the aerodynamic performance of a transonic single stage wide chord blade axial flow compressor through 3D steady state CFD analysis using commercial code ANSYS CFX.
- The analysis was carried out for various tip clearance configuration varying from 0% to 1.2% of rotor tip axial chord. The characteristics plots like corrected massflow rate versus pressure ratio and corrected massflow rate versus efficiency were generated for various tip clearances at design speed.
- The study concluded that the zero tip clearance is not yielding higher pressure ratio and highest efficiency. The optimized tip clearance value 0.3% of rotor tip chord at the design speed gives higher values of aerodynamic performance parameters like pressure ratio, efficiency and surge margin.
- The extent of aerodynamic performance deterioration due to manufacturing deviation of the components will be available prior to engine testing in the test bed using data generated in this work.,

ACKNOWLEDGMENT

The authors are thankful to the entire compressor group team for their support for making this work possible. The authors also thank the Director, GTRE for permitting to publish this work.

REFERENCES

1. Inoue, M., Kuroumaru, M., Yoshida, S., Minami, T., Yamada, K. and Furukawa, M., "Effect of Tip Clearance on Stall Evolution Process in a Low-Speed Axial Compressor Stage", ASME Turbo Expo, paper GT2004-53354, Vienna, Austria, 2004.
2. Domercq, O. and Escuret, J.-F., "Tip Clearance Effect on High-Pressure Compressor Stage Matching", J. of Power and Energy, 221, pp. 759-767, 2007
3. Xicheng Jia, Zhengming Wang and Ruixian CAI (2001) "Numerical Investigation of Different Tip Gap Shape Effects on Aerodynamic Performance of an Axial-Flow Compressor Stator", Proceedings of ASME TURBO EXPO 2001, New Orleans, Louisiana, USA, 2001-GT-0337.

4. Wang Songtao and Wang Zhongqi (2002) "The Tip and Hub Leakage Flow of a Repeated Two Stage Compressor", *Proceedings of ASME TURBO EXPO 2002, Amsterdam, The Netherlands, GT- 2002-30437*.
5. Gong Hee Lee and Je Hyun Baek (2002) "A Numerical Study on the Structure of Tip Clearance Flow in a Highly Forward Swept Axial-Flow Fan", *Proceedings of ASME FEDSM*.
6. W. Trevor John and P. Susan Prahst (2002) "3-D Digital PIV Measurements of the Tip Clearance Flow in an Axial Compressor", *Proceedings of ASME TURBO EXPO 2002, Amsterdam, the Netherlands, GT-2002-30643*.
7. Donghyun Y. et. al "Progress in Large-Eddy Simulation of a Rotor Tip-Clearance Flow", *12th Annual DOD HPCMP User Group, Conference, Austin, TX, 2002*.
8. L. Suder "Experimental and Computational Investigation of the Tip Clearance Flow in a Transonic Axial Compressor Rotor", *NASA TM-106711, 1994*.
9. Arnaud, D., Ottavy, X. and Vouillarmet, A., "Experimental Investigation of the Rotor-Stator Interactions within a High-Speed, Multi-Stage, Axial Compressor. Part 1 - Experimental Facilities and Results. Part 2 - Modal Analysis of the Interactions", *49th ASME Turbo Expo, Vol. 5A, pp. 903-924, Vienna, Austria, 2004*.
10. Reid, L., and Moore, R. D., 1978, "Design and Overall Performance of Four Highly Loaded, High-Speed Inlet Stages for an Advanced High-Pressure Ratio Core Compressor," *NASA TP 1337*.
11. B. R. Hutchinson, M. Ivanovic and E. M. Bennett., "Predictions of Transonic Compressor Flow," *Fourth Annual Conference of the CFD society of Canada, June 2-6, 1996*.
12. Kenneth L. Suder, "Blockage Development in a Transonic, Axial Compressor Rotor," *NASA TM 113115, 1997*.
13. Prashant, P. U. (2013). *Introduction to Novel Concept of Harnessing Mechanical Energy from a Collapsible bladder Connected to Flexible tubes. Preliminary Application in design of ultra, low head-low flow and efficient novel hydraulic turbine. Research and development (IJCSEIERD), 3(1), 17-30*.
14. Rodrick V. Chima, "SWIFT Code Assessment for Two Similar Transonic Compressors", *NASA TM2009-215520, AIAA-2009-1058*.
15. Ali A. Ameri., "NASA Rotor 37 CFD Code Validation, Glenn-HT Code," *NASA/CR-2010-216235, AIAA-2009-1060*.
16. Jabbar, A. A., Rai, A. K., Reedy, P. R., & Dakhil, M. H. (2014). *Design and analysis of gas turbine rotor blade using finite element method. International Journal of Mechanical and Production Engineering Research and Development, 4, 73-94*.
17. Hanoca P, Shobhavathy M. T "CFD Analysis to Investigate the Effect of Axial Spacing in a Single Stage Transonic Axial Flow Compressor", *Symposium on Applied Aerodynamics and Design of Aerospace Vehicle (SAROD) 2011, Bangalore, India*.
18. Kirubakaran, P. et. al. "Aeroelastic Flutter Investigation and stability Enhancement of a Transonic Axial Compressor Rotor Using Casing Treatment", *Vol.1: Compressore, Fans and Pumps, Turbines, Heat Transfer, Combustion, Fuels and Emissions, 2017*.